Bambu Lab FDM Training SOP

Training Checklist:

- Components Overview: print head, xy gantry, build tray,
- Safety: hot printheads & bed, moving components
- Job Setup:
    - At the computer: CAD → export .stl → Bambu Studio OR OrcaSlicer (print parameters, support materials) → select “Innovation Workshop X1C” printer → start print (make sure “Enable AMS” and “Bed Leveling” are selected)→ record in FBS
    - At the printer: remove leftover supports/calibration lines → clean build tray

Safety Concerns

- Both print heads and bed are heated during operation. Do not attempt to clean, remove, or adjust without allowing for adequate cool down time.
    - Wait until it cools down to approximately 60 C

Common Mistakes

- Inadequate part adhesion to the bed - can cause print failure or breakage of entire print head assembly
    - Can be prevented with proper bed cleaning, use of hairspray, bed adhesion print settings, and high enough printing temperature

Safe Operating Procedures Review

1. Launch Bambu Studio (green Bambu Lab logo) or OrcaSlicer
2. Start a fresh plate: select File → New Project
3. Import your STL file onto the software: select File → Import → Import 3MF/STL/… → choose your STL file
4. Set up scale, orientation, arrangement, and supports as needed
5. Select the right filament and the right nozzle
6. Slice plate and make reservation on FBS
7. Before starting print, scrape off any leftover support and calibration lines and clean build tray with soap and water
8. Align build tray in printer with the smaller tab facing the back
9. Check AMS to see if there is enough filament for your print. If not enough filament, replace the spool
10. Once build tray/printer is all set up, click “Print Plate” on Bambu Studio
    1. Select “Innovation Workshop X1C” for the printer
    2. Make sure only “Bed Leveling” and “Enable AMS” is selected and start print

Post Processing

- Remove part from print bed using your hands or a scraper (try not to touch the build tray with your bare hands)

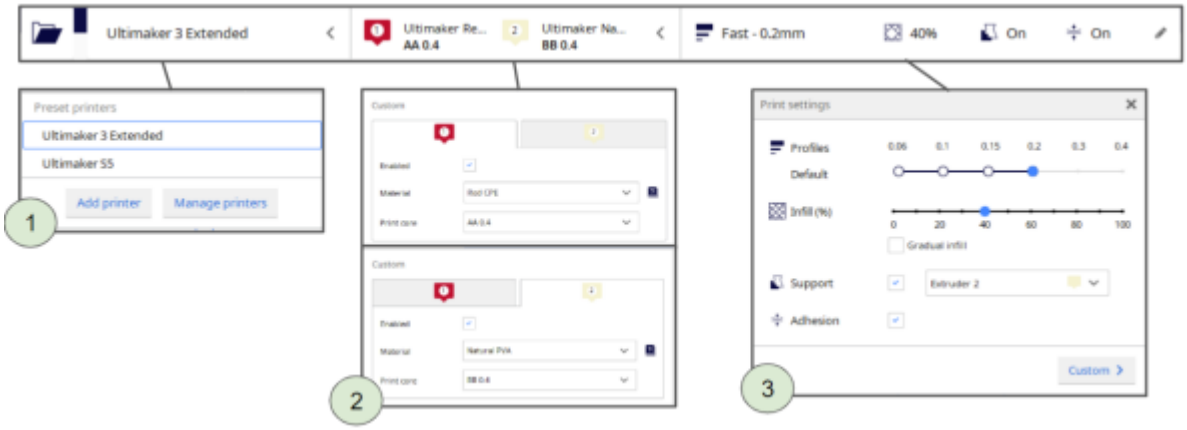
Maintenance

- Scrape off calibration lines on the sides of the build tray
- Bed should be cleaned with soap and water in the sink

Additional Information

Cura Model Settings

1. *Auto Orient*: Gives you the recommended orientation for your print. Usually is the best orientation, but you should still think about specific qualities of your print.
2. *Arrange*: Arranges your models near the center of the build tray
3. *Rotation*: Allows you to rotate your model. Make sure to select the face that you want to align to the print bed. Ensure that you align the face with the **greatest surface area.**
4. *Mirror*: Allows you to mirror your model.
5. *Per Model Settings*: Allows you to change print settings for each model. This allows you to have multiple prints on the same build plate while still using different print settings for each model.
6. *Support Blocker*: Allows you to prevent support from being generated in certain areas.
7. *Print selected model with Extruder 1*: Select which model to print with Extruder 1. This is generally build material.
8. *Print selected model with Extruder 2*: Select which model to print with Extruder 2. This is generally support material.



Cura Print Settings

1. *Printer Type*: Choose Ultimaker 3 or Ultimaker 5 Extended.
2. *Material Type*: This allows you to choose the material type you have loaded into the printer. Ensure that both the material type as well as the printhead diameter settings match the printheads on the printer
3. *Layer thickness, infill, support*: Allows you to adjust the thickness, infill, and support you wish to use. **Ensure that you enable adhesion**